Reply to reviewer A: Reviewer comments and adjustments made

Reviewer's comments helped to clarify many areas in the paper. The reviewer's efforts are greatly appreciated. I rephrased and explained many of the ideas according to the suggestions. The reasons for having many of these questions may be due to (1) the limitations on the length of the paper, (3) delay in the printing of my paper "Performance comparison of overland flow models.." in which I explained most of the test cased used in error analysis (2) somewhat different nature of the Hydrology in South Florida. Following are the reviewer's comments, my responses, and corrections made.

Comment: "On page 11 the author claims that only an upstream b.c. is needed.."

Reply: The reviewer is correct in that a diffusion model needs a downstream boundary condition as well as the upstream boundary condition. I said "only one boundary condition is needed..", and I didn't say that it has to be specifically an upstream boundary condition as mentioned by the reviewer. In general we think South Florida as a 2-D domain instead of a river reach which has a clear upstream and downstream boundary. Unlike when using full equations to solve 2-D subcritical flow which requires 2 upstream b.c. and one downstream b.c. s, when using diffusion flow, one is needed at the upstream, and one at the downstream end because only the head is solved in the diffusion equations. I will slightly modify the sentence in the paper to clarify the matter.

Comment: "On top of page 12, the author suggest replacing $\mathbf{M}^{\mathrm{n+1}}$ with.., Is this for the entire equation, or only for $\mathbf{P}$.."

Reply: The objective is to obtain the best value of $\mathbf{M}^{\mathrm{n+1}}$ by iterative means to satisfy the above equation. $\mathbf{M}^{\mathrm{n}}$ however is does not change during iterations. So, $\mathbf{M}^{\mathrm{n+1}}$ is changed during iterations. I am improving my explanation to avoid the confusion.

The method becomes explicit only when $\alpha = 0$ in which case $\mathbf{P}$ becomes diagonal.

When I mentioned "iterations were not used in the current application", I meant that I used exactly the method used by Akan and Yen (1981), and used the $\mathbf{M}$ of the previous time step, and didn't bother to update $\mathbf{M}$. The iterations listed in table 1 are not the iterations on $\mathbf{M}$ but iterations within the sparse linear solver. They indicate the computational effort in obtaining the solution of the linear equations. If iterations are carried out to improve $\mathbf{M}$, 2-4 iterations are needed at each time step (p-12). The number of sparse solver iterations are shown in Table 1.

Comment: ".. talk about the imposition of head boundary condition in eq (27). Since they modify the row elements to impose the b.c., the symmetry is lost"

Reply: True. The way head b.c. is applied is simple, but it destroys the symmetry. Surprisingly, we found out that nonsymmetric solvers in the SLAP (Lawrence Livermore Lab) and PetSc (Argonne National Lab) packages didn't take any extra time to solve these non-symmetric matrices. As a result, there was no immediate incentive for us to modify the method to make it symmetric. But for the sake of keeping the final matrix symmetric, we are trying to modify the matrix and make it symmetric. We are also trying to see if a penalty function approach can be used in the future as suggested by the reviewer, and keep the method is simple. However, none of these would affect the solution.

Comment: "Figure 3a and 5 are not symmetric..",

Reply: This is because the mesh was non-symmetric, and crude, only with 238 cells. The contour plotter program TECPLOT needed the interpolated nodal values and not the computed cell values for the plots. This interpolation also somewhat affected the final contour plot. With finer meshes, as it the case with 1536 cells in fig 6, the figures were symmetric, and the solution matched very closely as shown. By using an example with 238 cells, I was trying to demonstrate the nature of numerical errors, which won't be seen with too many (1536 in this case) cells.

Comments: "comparing fig 3a and 5 shows that the modflow solution is more accurate.

This makes me wonder why one would be interested in the method"

Reply: The modflow solution in the figure was obtained using a 40x40 discretization or 1600 cells, and the current method only used 238 cells. So, the comparison is unfair. On the other hand, a comparison with 1536 cells as shown in fig 6 is so close to see with the naked eye, because the lines coincide. I was trying to demonstrate the behavior of the solution, and the effect of low resolution grids at the same time. The biggest advantage of the current model when compared with MODFLOW is the ability to use non-rectangular grids. When both use implicit solution methods of the same type, models with comparable numbers of cells will have comparable run times and accuracies. When rectangular cells are used, the current method gives a MODFLOW like finite difference method.

Comments: "I think the author presents a new/different.., there is no further insight gained by the work"

Reply: The challenge in modeling in South Florida is to develop models that can study vast areas, such as the entire South Florida from Orlando to Miami, and Palm Beach to Tampa. There is a demand for detailed models that use fine polygonal grids describing urban areas. Such models are also expected to simulate thousands of large structures and canals in South Florida. The challenge is to do all the long term (20-30 yr) simulations in a Sparc-10-20 computer, and to do it now. Algorithm efficiency is the primary concern in the paper. Considering the complexity of the coupled overland flow/gw flow system in South Florida, it became important to use a disciplined structured approach for modeling too. What you see here are the initial results from that effort. Existing models used for the purpose run into time step restrictions, 2 mile x 2 mile grid limitations and 1-3 hr run times. The RBFVM-2D model (page 19 of the paper) is one existing model that takes days to run only a few hours of the the Kissimmee problem with 1-2 s time steps. The current model can use 100 s and larger time steps with the same problem, and takes only minutes to run (p-19). With the current method, more computational time is needed only in the case of significant flow variations, and computations can go faster at near-steady state (p-13, para-2).

With the finite volume approach, it has become easier to describe flows in terms of cell walls, and replace cell walls with a levees if necessary. Being able to explain the physics to the public (hopefully) was also a minor consideration in the selection of the finite volume method. I have modified the paper in many areas based on the valuable comments made by the reviewer.

Comment: I am further put off by his statement on top of page 15 where the accuracy was obtained to 4-5 decimal places.

Reply: The related paragraph will become more clear after my paper on error analysis comes into print after the publication queue in ASCE (HY). The test was used here to compare numerical errors at different resolutions. I used a test run for a standard problem at an extremely high resolution to compute a near-exact solution, and used that result to compute errors of models at lower resolutions. The 0.442105 m value in p-15 was obtained from the test using hours of computer time, and is used as the basis to compute errors in Table 1. Unless 6 decimal places are used, errors in the 4 th decimal place could not be captured. The alternative was to use different test case dimensions so that one does not have to use this many decimal places. But, considering that the typical dimensions used in the Everglades are of this magnitude, the test was confined to similar dimensions, and I was stuck with 5 decimal places in the error analysis. It does not mean in any way that the coarse resolutions used in actual models could give this many decimal places. It only gives the benchmark values used in the error analysis that need this many decimal places to obtain meaningful results in Table 1. The paper listed as "Performance comparison .." has a comparison of errors of a number of other models too, using the same circular test problem. This paper would clarify the numerical test more. I agree with the reviewer on the emphasis of the paper, and have rephrased some statements.

Comment: I was not able to understand the error definition on the bottom of page 15-it does not appear to be the standard norm. Defining the error based on one space-time point result does not constitute a valid test.

Reply: Most of the results on error analysis are related to my paper "Performance comparison of overland flow algorithms", and it may make more sense once the paper comes on the ASCE (HY). The idea for the test came from typical flood situations in South Florida and the Everglades. When rainstorms occur over relatively flat, shallow (1-3 ft) sheet flow, the net result is the creation of a water mound or a bubble that takes time to dissipate. The test was designed to mimic some of these conditions that the model is expected to simulate. In addition, the paper "Performance comparison.." has results that compare a number of overland flow model results with the same test, and therefore was used here. Even if the way the error was defined was uncommon, the method was consistently used with other models tool. I guess, that the situation is different in the Everglades from the rest of the country when you see miles of horizontal space around you, and the water level is only a few inches different between these points. The only thing you can compare the error at a given point, is against its local depth. The center is the location where the error is largest as well. This error has been used only for comparative purposes.

Comments: Statements on Page 9 and 19 ... Page 9 says if acute angled triangles are not used, the error is likely to be larger. Page 18 says that discharge estimates are off because acute angled triangles were used."

Reply: On page 9, I was referring to the circumference based method in which the numerical error increases when the triangle has an obtuse angle, and the circumcenter lies outside. We are currently trying to find a method that works equally well for non-acute angled triangles too. On page 18, I was referring to the results coming from line integral type walls in which $\bar{\mathbf{F}}$ is computed as the average of the nodes defining the wall. Ths line integral method from Hirsch's book (1989) was found to be inadequate because the average of nodal values does not give the best estimate of wall flux for acute angled triangles. The circumcircle idea came when reading about the work on mixed finite element method by Cordes and Putti (1996). This method is suitable for acute angled triangles. We haven't experimented much with obtuse angled triangles.

Comments: On page 26, Table 1, what is the meaning of the column that contains the number of iterations.

Reply: This is the number of iterations taken by the sparse solver. If the flow conditions have changed much from one time step to the other, or if there is rain, there will be many iterations. Under steady state, and no rains, there will be few. It indicates the computational effort.

Reply to reviewer B: Reviewer's comments and adjustments made

Reviewer's comments were useful in understanding the areas that are not sufficiently explained in the paper. One example is the significance of the method for use in the Everglades. There were many ideas in the technical area as well. I am planning to use the suggestions made by the reviewer as stated below, and also to improve the general discussions.

Comment: In the end, I was disappointed in the results since the model was not demonstrated on a problem that was of significant enough difficulty..

Reply: I can understand the reason, and I am addressing it in the writeup. The model is to be applied over the entire South Florida and the Everglades, with large natural, agricultural and urban areas requiring a variable discretization. The purpose of the paper is to test the model in a small, well tested Kissimmee area to see the performance. Kissimmee case has actual field data, and some physical and numerical model data from the University of California at Berkeley. The numerical model results are published in Zhao and Shen (1994), with the description of the model RBFVM-2D. The purpose of the paper is to discuss the testing of the model using the current overland flow case before applying to the South Florida system with integrated canal network and ground water modules. In the paper, I have demonstrated that by being able to run with time steps as large as 20 sec (SLAP2.0) as opposed to 1-2 s in the case of RBFVM-2D for the same Kissimmee test problem, the current method is much more efficient. Simulation runs for Kissimmee with RBFVM2D are known to run for days in the Water Management District computers.

Comments: The method assumes that flow is normal to the cell boundaries. This is often not the case and this can significantly affect the numerical results of the simulation. This may be why some of the model results are not as accurate as desired (p-18).

Reply: In the mixed finite element method that is shown to be equivalent to the current finite volume method (by Cordes and Putti, 1996) the basis function uses the discharges across the three walls. It does not mean that the flow is normal to the wall. The discharges Q_1, Q_2.. are scalars. The use of Q_1, Q_2.. in the basis function is similar to the use of nodal heads in many finite element schemes. Each of the discharges, Q_1, Q_2, Q_3 in combination with the geometrical shape, produces the vector velocity in eq (21), which was originally derived in the references Raviat and Thomas (1977), and Cordes and Putti (1996). In my comment in p-18 "$\bar{\mathbf{F}}$ in (13) does not provide a very accurate estimate of the discharge", I was referring to the *line integral based method* as shown under the subtitle and not the circumcircle based method used by Cordes, et al. I found that local numerical errors can become a problems with the line integral base method when using acute angled triangles, even if the overall solution holds good. This is the reason why I recommend using the circumference method to avoid these problems.

Comment: Xanthopoulos and Koutitas, who were referenced, improperly convert from the two-dimensional.."

Reply: The reviewer is correct. The names Xanthopoulos and Koutitas were mentioned only to give credit to their early work on diffusion flow. There results were faulty because of the stated reason. With this mistake, it is not possible to obtain circular flow patterns in the test simulation even when the initial condition gives circular flow patterns. I tested this by carrying out numerical experiments with circular flow patterns. I found out that at high resolutions, both current methods (with circumcircle and line integral walls) gave perfect circles.

Comment: The author does not deal with anisotropic roughness. In the type of system involved in the Everglades..

Reply: The reviewer is correct. There can be many areas with anisotropic roughness in the Everglades. The reason why we have not included it in the model is because we do not yet have anisotropic roughness data yet. We don't even expect them in the near future. This is a good point to raise with the USGS teams that are studying the roughness in the

Everglades. In the mean time, I take the reviewer's suggestion and adjust the wording in the paper to reflect this fact, and make modifications to future model versions so that this effect can be be taken into account when data is available.

Comment: On page 8, line-integral method, the author needs to state how K_j are computed.

Reply: K_j is computed in eq (7) and (8) for node j. The nodal values of depth and Manning roughness are computed as weighted averages of surrounding cells. The weighings used are the areas of the surrounding cells. Hirsch's book uses the same method.

Comment: The convergence results on p-12 (2-4 iter) are inconclusive for general problems. ..

Reply: True. I just mentioned them to give a general idea, using the present case as an example. I have not seen iterations being carried out with diffusion flow models anywhere. I don't think they are needed in most cases. Number of iterations can be reduced by using smaller time steps. For the cases I studied, it was hard to find one that needs iterations. If it is really rapidly varying flow, iterations may not help, and one may need to go for dynamic modeling instead of diffusion flow modeling. The fact that the number of iterations is shows that it doesn't take many iterations even if one wants to iterate.

Comment: The three examples are;..., In our experience, real problems occur with unsteady flow on a highly irregular..

Reply: The test cases were limited due to the length of the paper. The steady state problem with the complex geometry was selected because the same test that was carried out with the unsteady flow model RBFVM-2D (Zhao and Shen, 1994), and the data set was freely available. There were some slowly varying unsteady data as well for Kissimmee; but this data did not demonstrate anything more because the flow was too slowly varying.

The current model is finally to be used to simulate vast landscapes in South Florida and the Everglades in which the flow is shallow, slowly varying, and solvable using diffusion flow methods. Many of the land use patterns in South Florida are polygonal, and not rectangular. We do not intend to apply the current model over deep lagoons or even lake Okeechobee. The velocities we intend to study are in the range 0.001-0.01 m/s, whereas in the Kissimmee test case, the model was tested with 1 m/s velocities. Having used dynamic models before, I was surprised to see how much of numerical problems go away when a diffusion flow model is used instead of a dynamic model, if the conditions allow you to do this. There are many cases in which this can be done. My experience with the Niagara River near Buffalo, NY using a 2-D dynamic model reminded me of the reviewer's concern. But if we are only interested in slowly varying flows, I will not hesitate now to use diffusion flow models for Niagara as well, considering how much of "numerical" pain they can avoid. In the case of the entire South Florida Region which is far more complex and large, the occurrences of extremely dynamic regional flow is rare, and diffusion flow models (just like ground water flow models) provide a more numerically friendly algorithms to use.

Comment: Based on our experience, I would expect this model to exhibit stability problems when unsteady flow is modeled in a complex floodplain.

Reply: Model results were present for the steady state case only because this is the case for which a good data set was available. The steady state was arrived by running the unsteady flow model for a very long time. Contrary to the statement, the contribution of the work is a model that is stable with very large time steps. I have found that the model is stable with time steps 50-100 times as large as the maxinum allowed for explicit models. This affects the run time in a significant way.

Marked item p-9 When acute angles are not used, numerical error can be large with the circumcenter based method. This is because the circumcenter, which is supposed to represent the triangle falls far outside the triangle with obtuse angled triangles. As a result, the flow patterns in such cases may get somewhat distorted, and the formulations get affected by error terms. Extension into extremely obtuse angled triangles is one area we plan to work on. Fortunately, the tin generators we have now can prevent giving obtuse angled triangles altogether, and this is not a pressing problem.

Reply to reviewer C: Reviewer comments and adjustments made

I gathered some very valuable information as a result of the review, including some references. Strelkoff's paper for example explained what I already observed in the model. This review helped me greatly to improve the paper. I modified many areas as suggested.

Comment: Post drainage features.. requires the reader to guess at its meaning...

Reply: I changed the word to "present day Everglades" because it is the word used at the District now. I improved the rest of the sentence too.

Comment: The definition of "circumcenter"

Reply: I added and modified the sentence. Even with the restrictions on the length of the paper, I tried to add more information. I am taking the reviewer' advice on the paper being able to "stand alone".

Comment: p-4 second paragraph: Does the author mean a numerical analysis of .."

Reply: I meant an analysis of numerical errors, and changed the text accordingly.

Comment: p-5, following eq (4): neglect of the local acceleration term while retaining the convective acceleration term..

Reply: The reviewer is correct in the statement. The intention of neglecting the local acceleration and retaining the convective acceleration was to improve rapidly converging and diverging flow solutions near boundaries. I found out the hard way, that it does not work for regular unsteadiness, by observing the results. I thank the reviewer for pointing out the reference Strelkoff, Schamber and Katopodes (1977). I already have a description of this in the paper.

Comment: p-6, eq (7) did the author intend n_b here, along with general γ and λ?

Reply: No. The reviewer is correct, in that n_b is not Manning coefficient any more.

Comment: p-6 following eq-8, what does the author mean by "more continuous flow"..

Reply: When $K = 0$ was used instead of $K = K_0$ (a non-zero number), I saw a very small discontinuity in the flow and the head time series at these head values. A very small head difference remained between cells as a result even after level pool computations. This problem disappeared with a value of K_0 that was computed by setting a lower bound on S_n to avoid the singularity. The entire problem at small S_n is due to the numerical way of handeling a singularity, and fortunately does not affect the model results in a detectable way.

The reviewer is correct in that $h = 0$ is a dry cell. There is nothing more needed to facilitate it. The statement is improved to reflect this now.

*Comment:*And it is not clear how K linearizes the equation except for...

Reply: K is used to linearize the equation by assuming that K of previous time step remains constant during the H computations current time step. $\mathbf{M}$ of eq (26) is made of these K values. The iterations mentioned below eq (26) is intended to capture some nonlinearity if possible. But Akan and Yen (1981) never used it. Introduction of K_0 whether it is zero or not, is a way to avoid the singularity as stated by the reviewer.

Comment: The author's volume integrals are really surface integrals.

Reply: Correct.

Comment: p-8 Eq (14) is an unusual application of the Gauss's transformation..

Reply: Correct. Not commonly used, but elegant. I saw it for the first time in Hirsch's book. I am adding some explanation.

Comment: The term "shadow polygon" should be introduced...

Reply: This was a term used for the polygon made from the centroids of the main polygons. It looked like a good term for this second set of polygons. I will improve the explanation.

Comment: p=8, eq (15): How are nodal values K_j calculated?...

Reply: They are computed using eq (7) and (8). Nodal values of n_b, h are computed as weighted averages of the surrounding cell values. The weighted averages are computed using cell areas as weights. I am improving the text by adding the explanation. I found the averaging method in Hirsch's book.

Comment: The arrow over the first H is a typo. .. when p=1, p-1=0 etc..

Reply: Yes. The idea of p is to carry out line integration around the polygon in a complete closed manner. Thanks for pointing this out. I am correcting them.

Comment: p-9 eq (18): for computing K_r the author suggests eq (7) or (8), with an average (weighted?) depth. This could lead to nonsensical simulated flow..

Reply: The combinations of head conditions leading to physically invalid (nonsensical) were avoided by adding the necessary *if* conditions in the program code. The reviewer is correct in the comments. I was not sure as to how much of this "if" information could be presented in the paper, because a typical code has too many.

Comment: p-9 eq (21) The origins and meaning of this equation are not clear. What are Q_1, Q_2, etc....

Reply: True. $Q_1, Q_2, ..$ are scalar discharges across walls. A vector is made because of the vector quantities coming from the geometry are multiplied by $Q_1, ...$ This equation is a by-product of the original derivation by Raviat and Thomas (1977) for the mixed finite element method. In this method, the basis functions are derived using $Q_1, Q_2..$ instead of the usual nodal heads. Raviat and Thomas's book has a number of basis functions that could equally well can be used. Cordes and Putti (1996) showed that the mixed finite element method with this basis function gives the finite volume method used. This also implies hat the type of equation for $\vec{v}$ can be extended to the finite volume method as well, even if the finite volume method does not have basis functions. Since (21) is just an equation for interpolation of $Q_1, ..$, it is capable of describing unsteady flow computed in terms of $Q_1, ...$

Comments: p-13 The last paragraph before ... What does "starting fresh" mean?

Reply: This is a concept we try to exploit in the model. In traditional linear equation solvers, (elimination methods for example), when the linear equations are solved at every time step, one has to go through exactly the same elimination steps in the first, second, etc upto the last time step, eliminating row at a time, or whatever. Even if the first and the second time steps are exactly the same, or may be with only a slight rainfall in one cell, the computer has to repeat the elimination process time after time. The new methods don't have to do that any more. They can keep the matrix in the memory, and change only the few rows or columns with the rainfall in a quick operation, and produce the solution, without repeating the solution procedure. In the Everglades, this can improve things significantly. The model may take a relatively longer time during rainy season, but may run much faster during dry times, carrying out just the minimum necessary computations. Starting fresh means assuming that we do not use the previous time step solution to update, but begin fresh with nothing known about the system as explained.

Comments: p-36 Figure 8. The impossible results of velocities directed in two and even 3 directions..

Reply: This is an artifact of the grid used in the RBFVM-2D model, which we used too, for comparison purposes. The arrows were drawn at the circumcenters of the triangles in the case shown. For the grid used, the circumcenters sometimes almost coincided, creating the problem. Some of the cells are very small, creating many arrows in finely gridded fast flow regions. I had to make the arrows long enough so that small velocities become visible. This made arrows much bigger in size compared to their cells. I had to make the arrows thick enough so that reduction during publication become possible. The unfortunate result is what you see. The package used was TECPLOT 7. I am making improvements to clear the confusions.

KEY WORDS

1. Implicit finite volume method
2. Diffusion flow
3. External sparse solver
4. Everglades
5. South Florida Regional Simulation model
6. Hydrologic model
6. Hydraulic model
7. Computational hydraulics